International Communications in Heat and Mass Transfer 82 (2017) 114–124

Contents lists available at ScienceDirect

International Communications in Heat and Mass Transfer

journal homepage: www.elsevier.com/locate/ichmt

Numerical investigation on influence of diffuser vane height of centrifugal pump

Lei Fu, Xiangyuan Zhu, Wei Jiang, Guojun Li *

Key Laboratory of Thermo-Fluid Science and Engineering of MOE, School of Energy & Power Engineering, Xi'an Jiaotong University, Xi'an 710049, China

ARTICLE INFO

Available online xxxx

ABSTRACT

Vaned diffusers are extensively used in centrifugal pumps, but the influence of vane height on internal flow field and overall performance is not explicit. This paper mainly presents numerical investigation on influence mechanism of diffuser vane height in a single-stage centrifugal pump. The head values were carried out on a low specific speed centrifugal pump equipped with different diffuser vane height by numerical simulation and experimental method. And the deviation between numerical results and experimental results were <5%. The diffuser vane height h/b ratio is changed as 0, 0.3, 0.4, 0.5, 0.6, 0.8, and 1 in this study. The numerical analysis shows that reducing diffuser vane height could eliminate the vortex which appears at tongue region. Meanwhile, the influence of rotor-stator interaction was reduced by reducing the vane height. Consequently, the energy loss in the volute and the diffuser could both be decreased at design flow point and over flow point. In the other hand, the circumferential velocity at partial flow point gets larger which could lead to large frictional loss. In general, reducing the diffuser vane height at design and over flow point could improve the output work of impeller.

1. Introduction

Diffusers are extensively applied to centrifugal pumps with the function of converting the exit kinetic energy of impeller to static pressure recovery, and decrease the radial thrust of impeller in single-stage centrifugal pump especially. Recently, the influence of vaned diffuser on flow patterns in centrifugal pump has been studied widely by means of experiment and numerical simulation.

Feng et al. [1–4] had published numerous works about unsteady inner flow field in centrifugal pump with vaned diffuser, either using CFD or visualization (PIV, LDV) techniques. The works primarily target several turbulence effects and other unsteady features attributed to the rotor-stator interactions between impeller and diffuser in the design and off-design conditions. Manish Sinha et al. [5–7] investigated the flow structure and turbulence during developed state of rotating stall in a centrifugal pump with vaned diffuser via PIV and LES techniques. The results show under stall conditions, when the low-pass-filtered pressure is high, the flow in the diffuser passage will jet outward; when the filtered pressure is low, a reverse flow will appear. In addition, relevant reference material on rotating stall, flow structure and turbulence in pump with vaned and vaneless diffusers can be found in H Stel et al. [8], Masahiro Miyabe et al. [9], G Wuibaut [10], F. Shi [11], Alfred Fontanals [12], Akinori Furukawa [13], Olivier Petit [14], Willian Segala [15], G Cavazzini [16] and M. Zhang [17] as well as Atif Abdelmadjid [18] and A. Akhras [19]. Recently, there is a new type of diffuser (diffuser vanes with reduced vane height) called half vane diffuser is proposed to improve the static pressure recovery of high specific speed centrifugal compressors by Yoshinaga et al. [20].

Operating range of centrifugal pump will be reduced by using vaned diffuser and static pressure recovery will be low at off design conditions in centrifugal pumps, although the vane diffuser can give higher static pressure recovery near the design condition [21]. In the other hand, the study on action mechanism and the influence of diffuser vane height on inner flow field in pump is not sufficient. The present investigation was conducted numerically for action mechanism and influence of reducing vane height on centrifugal pump operating performance.

2. Geometry and numerical method

2.1. Numerical method

The present investigation was carried out with a centrifugal pump consists of an impeller with 6 rotor blades, a volute casing and a diffuser with 5 blades. The design speed of the pump is 2900 rpm; the design volume flow rate and the design head respectively are 40 m^3/h and 60 m. The schematic layout of the centrifugal pump with vane diffuser is shown in Fig. 1.

The diffuser vane height is varied for different cases in numerical simulation. In present investigation, the value of h/b is taken as 0, 0.3, 0.4, 0.5, 0.6, 0.8, and 1.0. The diffuser with vane height of h/b of 0 is

* Corresponding author.
E-mail addresses: leifu@mail.xjtu.edu.cn (L. Fu), liguojun@mail.xjtu.edu.cn (G. Li).

http://dx.doi.org/10.1016/j.icheatmasstransfer.2017.02.014
0735-1933/

Nomenclature

H	Head [m]
H_{loss}	Total pressure loss [m]
Q	The flow rate [m^3/h]
Z	The number of blades
W	Relative velocity [m/s]
p	The static pressure [Pa]
n	The rotating speed [rpm]
u	The circumferential velocity [m/s]
η	The pump efficiency [%]
b_2	The outlet width of the impeller [10^{-3} m]
b	The outlet width of the diffuser [10^{-3} m]
t_0	Starting time for one period of transient simulation
t	Rotating time [s]
s	Unit for second
D_2	The outlet diameter of the impeller [10^{-3} m]
D_3	The inlet diameter of the diffuser [10^{-3} m]
D_4	The outlet diameter of the diffuser [10^{-3} m]
D_5	The inlet diameter of the volute [10^{-3} m]
Q_{des}	The design flow rate [m^3/h]
n_s	Specific speed [rpm,m^3/s,m]
V_r	The radial velocity of a certain point in the impeller [m/s]

vaneless diffuser and of h/b of 1.0 is vaned diffuser. The specifications of the pump stage are illustrated in Table 1 in detail.

To take leakage flow effects into account, the side chambers are also included in the grid generation. Then the computational domain is comprised of the inlet section, impeller passage, front side chamber, back side chamber, diffuser passage and volute casing with outlet section. The lengths of the inlet section and outlet section are large enough in 3-D model to assure the inner flow field relatively stable in simulation. The flow domain is divided into structure grid by using commercial software ICEM-CFD and the grid view of the computational domain is shown in Fig. 2, where the grid of impeller and diffuser are demonstrated for single passage.

Table 1
The specifications of the centrifugal pump stage.

Impeller outlet diameter D_2/mm	223
Impeller outlet blade height b_2/mm	8
Impeller blade number Z	6
Diffuser outlet diameter D_4/mm	283
Diffuser inlet diameter D_3/mm	228
Vane number Z_1	5
Design flow rate Q_{des}/m^3/h	40
Specific speed n_s/rpm, m^3/s, m	52
Rotating speed n/rpm	2900
Head c	60

For impeller and diffuser that are axisymmetric, only one blade passage needs meshing for defining a periodic connection between the two lateral sides of a certain blade and then the grid for other blade passages can be obtained by rotation around the rotating axis of the pump directly. Mesh refinement is defined at the region near tongue and walls to obtain more accurate flow structure details at these zones.

To guarantee the near-wall nodes within the log-law layer but the laminar sublayer which enables better resolutions on the boundary layer by the wall functions, the value of y + dimensionless distance between the first node and the wall is taken <60. The total number of grids for computational domain with different diffusers is no <4.82×10^6 (number of grids for domain with vaned diffuser), and the grid independence has been validated.

Commercial CFD software ANSYS-FLUENT was used for three-dimensional, unsteady incompressible Reynolds-averaged Navier-Stokes equations to analysis the inner flow field in a second-order algorithm via finite-volume method used for governing equations discretization scheme. Time-dependent terms were discretized by Second-order upwind scheme. Shear stress transport turbulence model with automatic near wall treatments was applied to simulate turbulence.

For steady and unsteady simulation boundary conditions, the total pressure condition is given at the inlet, and the mass flow rate condition is appointed at the outlet. The wall conditions of computational flow domain boundaries at entire domains are specified as no-slip wall

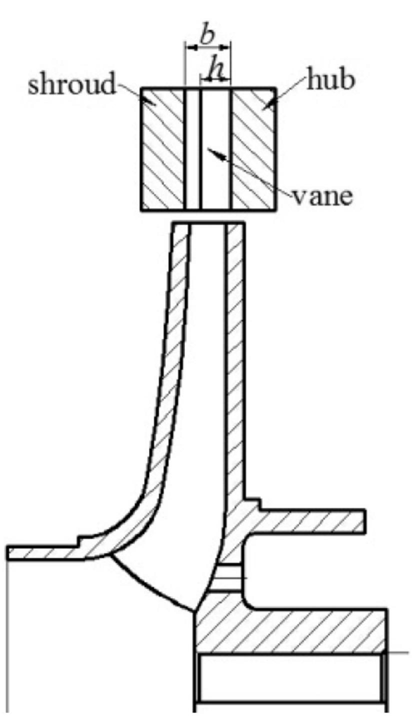

Fig. 1. Schematic layout of the centrifugal pump with hub vane diffuser.

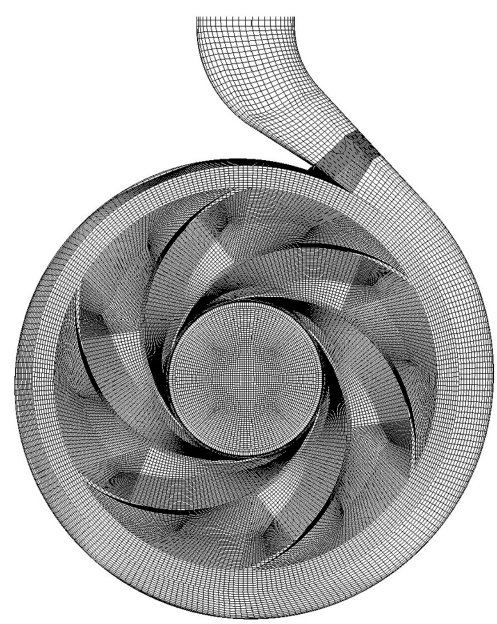

Fig. 2. Grid view of the computational domain.